Alterra GmbH
tel +41 44 825 05 55
www.geo-soft.ch info@geo-soft.ch

Manual

Alterra 1.2.2

Program for FEM geotechnical calculation

2012

Alterra GmbH can changed this document without prior notification.

Any part of this document connot be reproduced or transmitted in any form by any ways for any purposes without written agreenment with Alterra GmbH

© 2012 Alterra GmbH.
All rights reserved

Contents

Introdution 4

Material Models 5

1 Linear elastic model 5
2 Elastoplastic (Mohr-Coulomb yield criterion) material model 5
3 Double hardening model 5
4 Elastoplastic (Mises yield criterion) material model 6

Software overview 7

1 Main menu 8
2 Ribbon 12

Example #1 18

1 New project 18
2 Open project 19
3 Grid option 21
4 Point adding 22
5 Point deleting 23
6 Line creating 24
7 Line deleting 25
8 Area creating 25
9 Area deleting 26
10 Physical-mechanical properties of materials 27
11 Cross-section properties 28
12 Material property specifying 29
13 Mesh creating 32
14 Boundary conditions 33
15 Calculation 34
16 Result output 35

© 2012 Alterra GmbH

Introdution

Alterra is a program for geotechnical analysis based on finite element method.

The main feature of Alterra is calculation of "ground-base-structure" system mode of deformation and strength and stability analysis in plane or axisymmetric formulation.

The program is developed for solving difficult geotechnical problems (construction, maintenance and reconstruction of structures).

Three different material models for soil and elastoplastic (Mises yield criterion) material model for steel are used in this program.

Program features:

- Complex simulation of combined action of "ground-base-structure" system
- Stress-strain behaviour analysis taking into account plastic strains
- Staged construction and excavation
- Retaining wall analysis
- Internal CAD-editor or DFX-file import
- Computation of settlement and mode of deformation of structures
- Analysis of creeping landslide
- Analysis of internal forces in anchors and struts
- Analysis of subsidence trough
- Analysis of pile strength and bearing capacity.

© 2012 Alterra GmbH

Material Models

Three different material models for soil and one material model for steel are used in this software:

1. Linear elastic model
2. Elastoplastic (Mohr-Coulomb yield criterion) material model
3. Double hardening model
4. Elastoplastic (Mises yield criterion) material model

1. Linear elastic model

The most simplified and the most popular material model with linear relationship between stress and strain is defined by two values:

E - modulus of elasticity [Pa]

ν - Poisson ratio

2. Elastoplastic (Mohr-Coulomb yield criterion) material model

Elastoplastic model using parameters which are defined during soil laboratory testing.

The parameters:

c - cohesion [Pa]

ϕ - angle of internal friction [°]

E - modulus of deformation [Pa]

ν - Poisson ratio

k - discharge factor

3. Double hardening model

Double hardening model is elastoplastic model which uses parameters defined during soil laboratory testing and "pressure - deformation" curve defined during soil compression testing.

© 2012 Alterra GmbH

This model is based on the principle of stress and strain behaviour increase at hydrostatic pressure increase.

The parameters:

c - cohesion [Pa]

ϕ - angle of internal friction [°]

ν - Poisson ratio

λ, P_0 - hardening curve fitting parameters

k - discharge factor

$\varepsilon_v = -\lambda \ln((-\sigma_v + p_0)/p_0)$

4. Elastoplastic (Mises yield criterion) material model

Elastoplastic (Mises yield criterion) material model for steel.

The parameters:

E - modulus of elasticity [Pa]

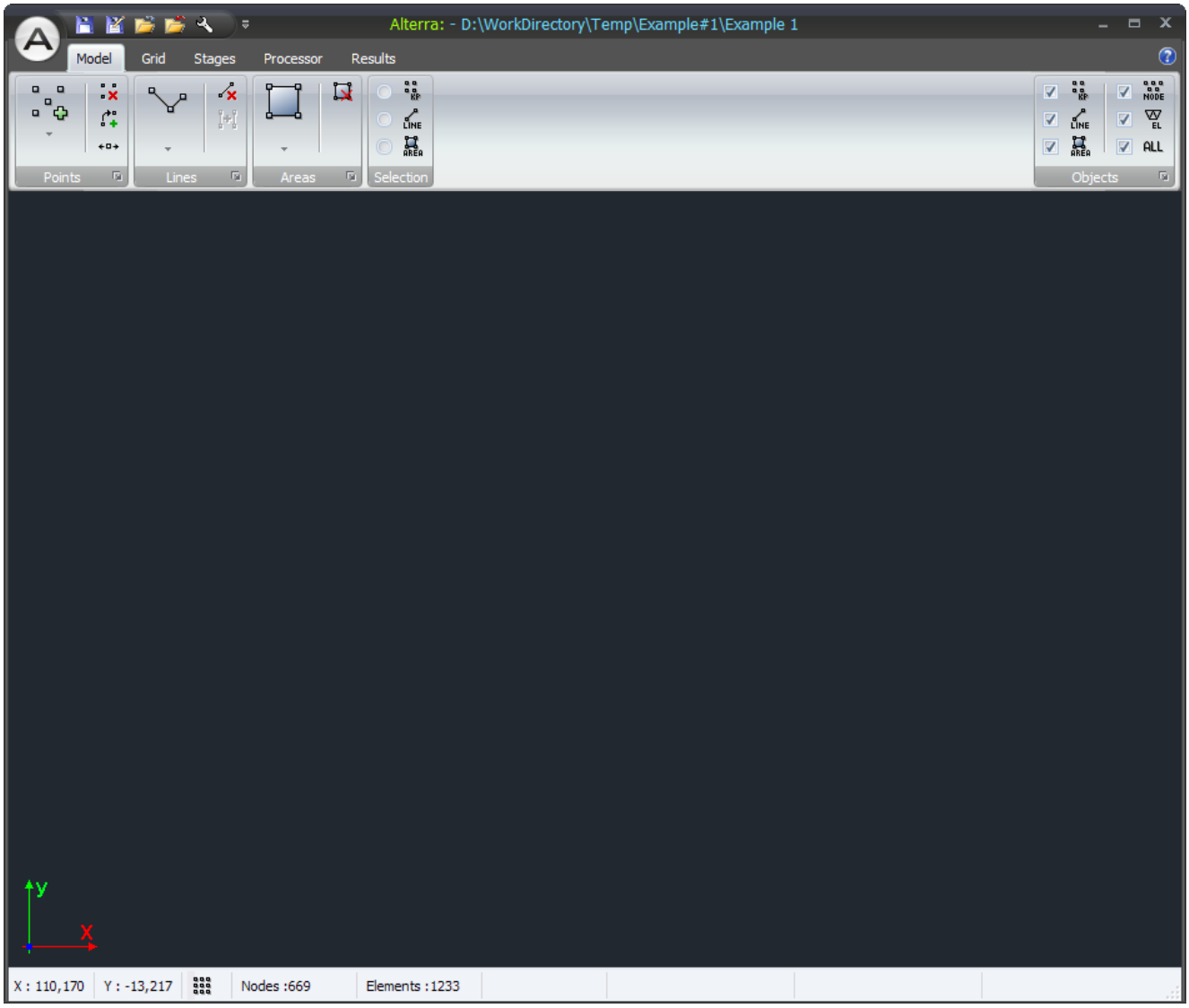
ν - Poisson ratio

σ_p - yield point [Pa]

© 2012 Alterra GmbH

Software overview

Main form has the following view:



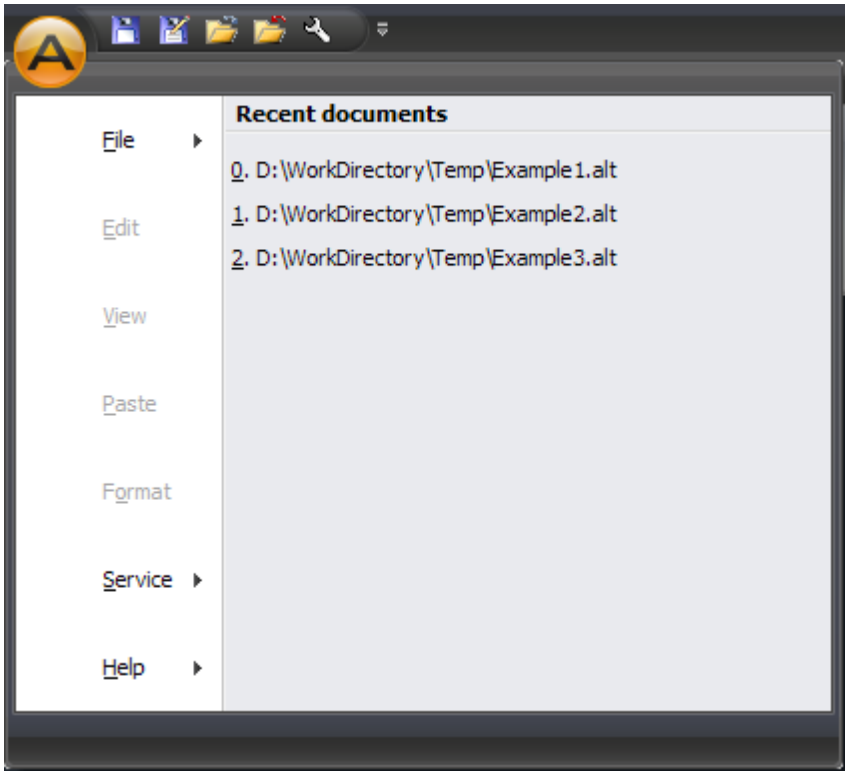
Main form consists of the following elements:

- Main Menu
- Ribbon
- Input field

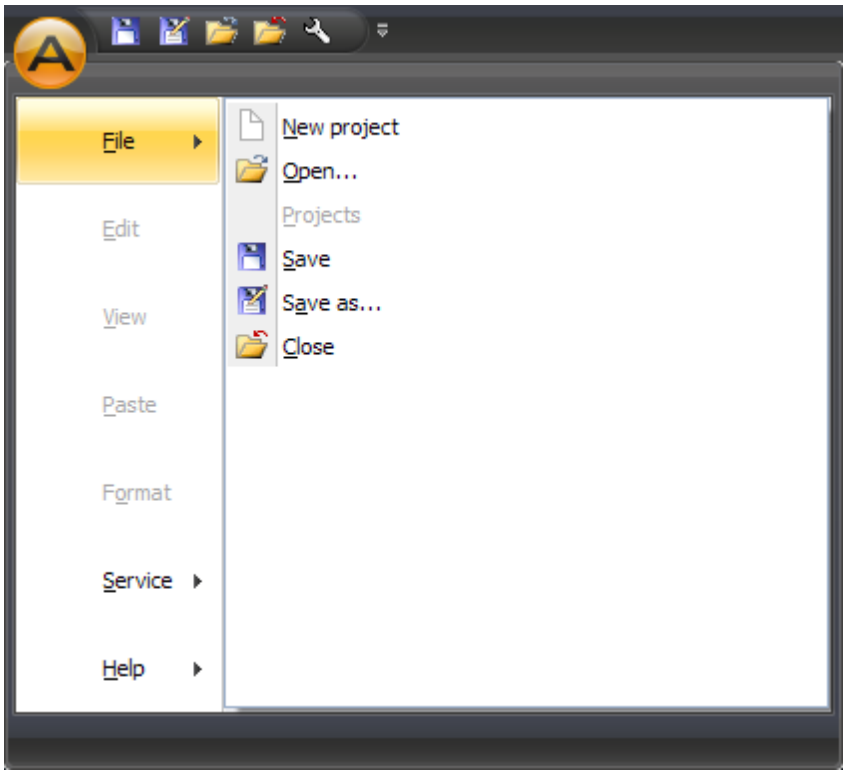
© 2012 Alterra GmbH

1. Main menu

Main menu consists of the following points: "File", "View", "Service", "Help".



"File" submenu:



New project – creates a new project (Ctrl + N).

© 2012 Alterra GmbH

...

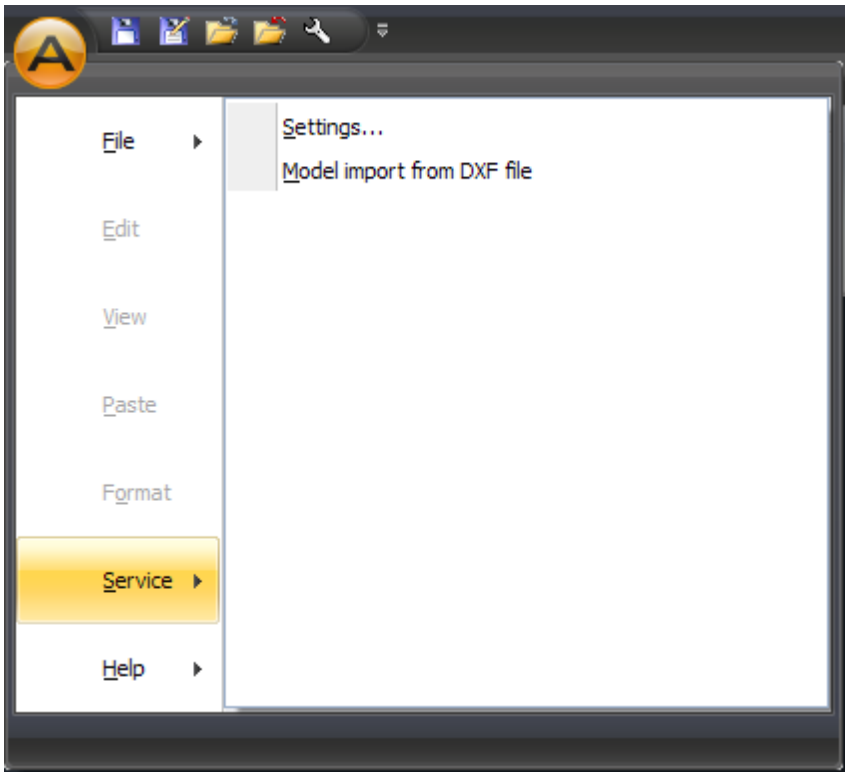
Open – opens a dialogue box of previously created project (Ctrl + O).

Save – saves the current project (Ctrl + S).

Save as – opens a dialogue box of current project saving under a new name.

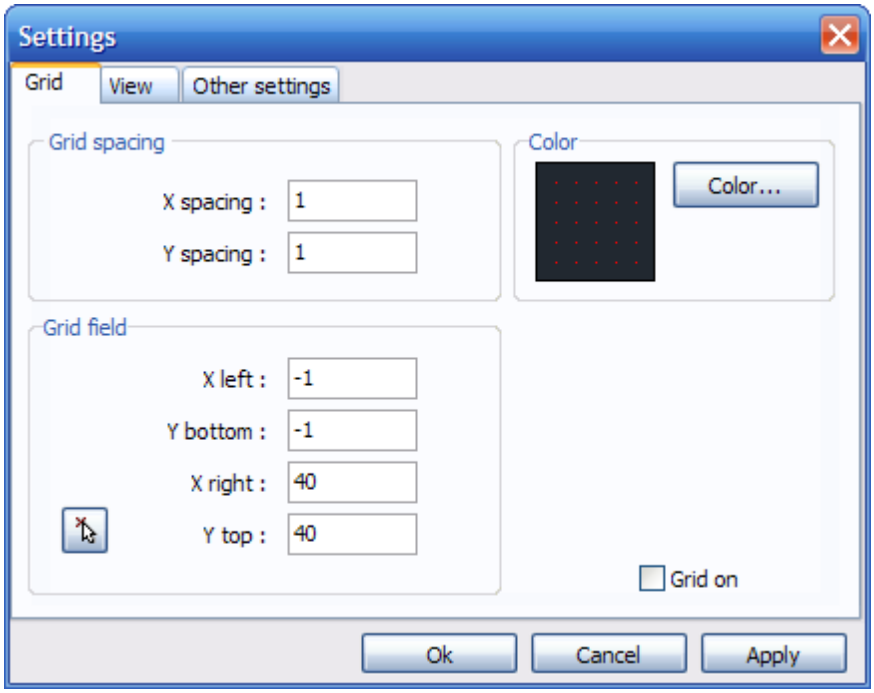
Close – closes a current project.

"Service" submenu:

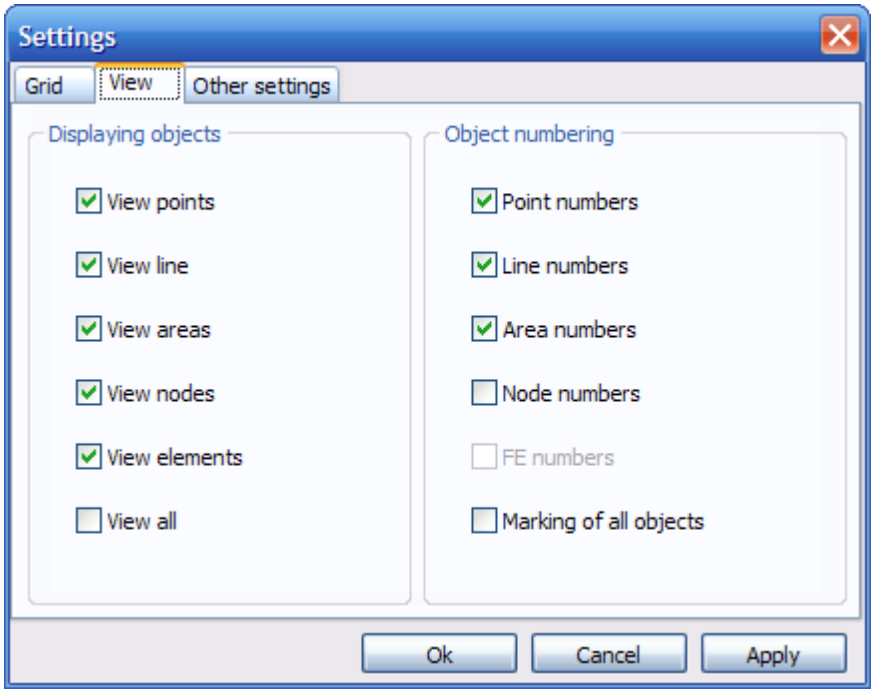


Settings - opens a settings box.

© 2012 Alterra GmbH



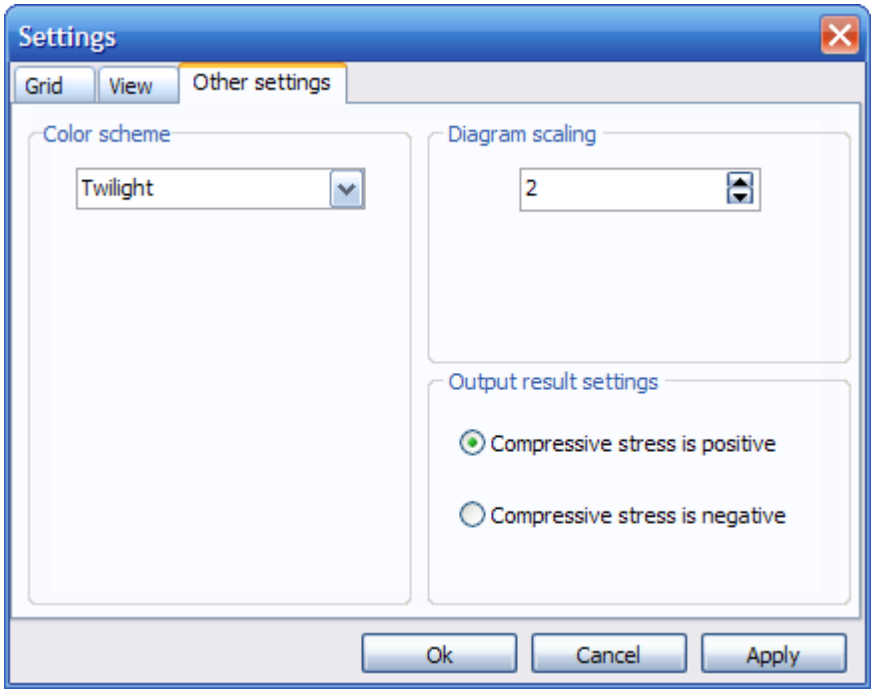
Grid option settings.



Design model visualization settings.

© 2012 Alterra GmbH

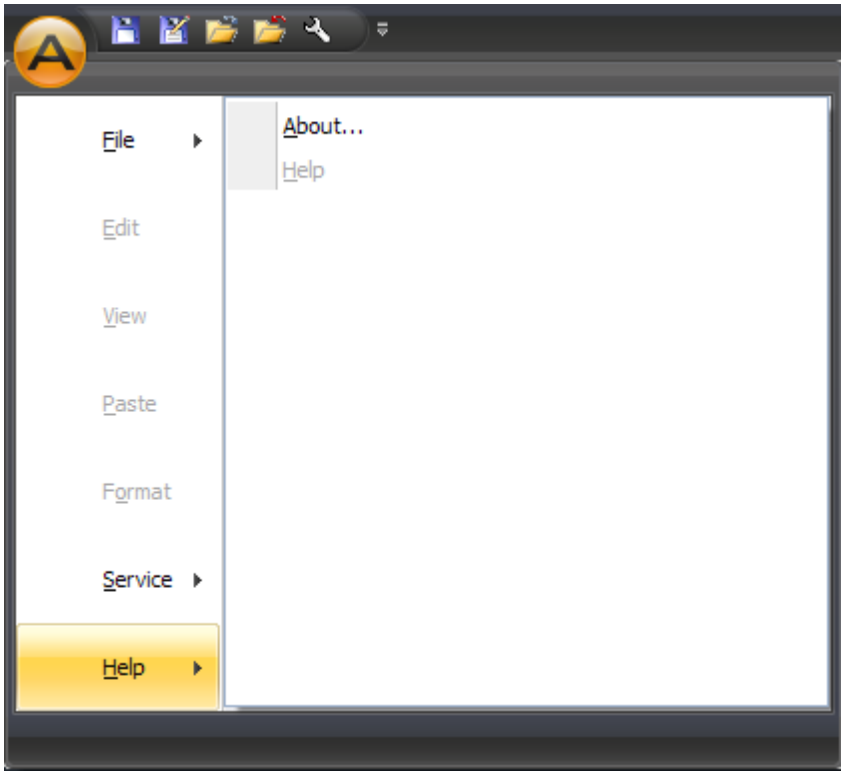
...



General option settings.

DXF file import – it is necessary that DXF file contains only points and lines (not polylines).

"Help" submenu:



Help – shows user manual (F1).

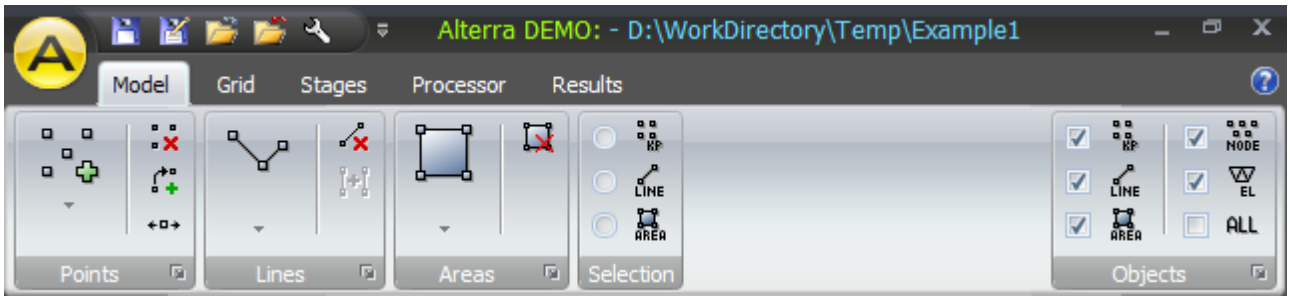
About – shows information about software version and developer contacts.

© 2012 Alterra GmbH

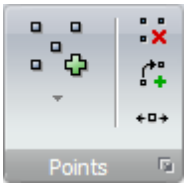
2. Ribbon

Ribbon consists of the following "pages":

1. Modeling

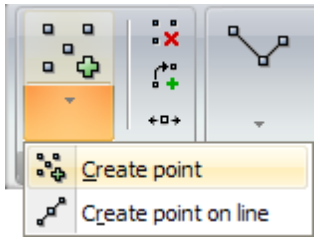


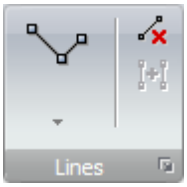
The page contains the following button groups:



– *Points*, allows creating, deleting, copying or moving points.

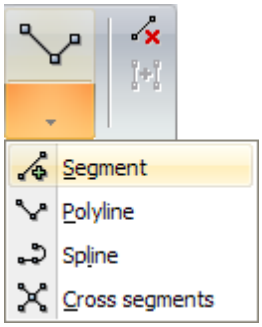
"Create point" button contains different methods of point creation:



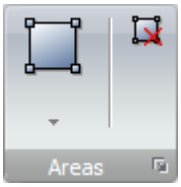


– *Lines*, allows creating or deleting lines.

"Create line" button contains different methods of line creation:

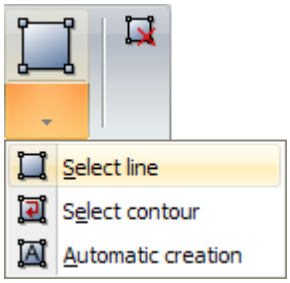


© 2012 Alterra GmbH

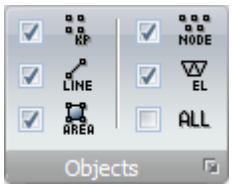


– *Areas*, allows creating or deleting areas.

"Create area" button contains different methods of area creation

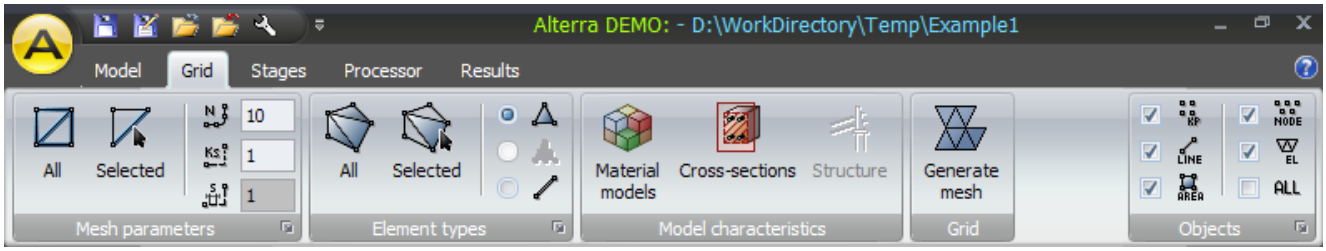


– *Selection*, defines which objects (points, lines, areas) are to be selected by "frame".

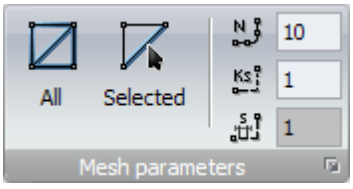


– *Objects*, allows quickly accessing to visualization options.

1. Mesh



This page contains the following button groups:



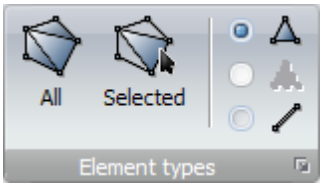
- *Mesh parameters*, allows specifying discretization parameters. These parameters can be specified for a certain line or for all lines.

Discretization parameters:

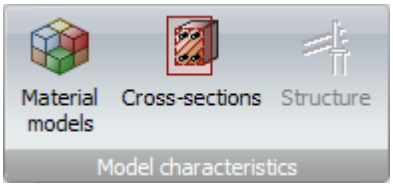
- node numbers in line and node concentration ratio

© 2012 Alterra GmbH

- element fixed size in line

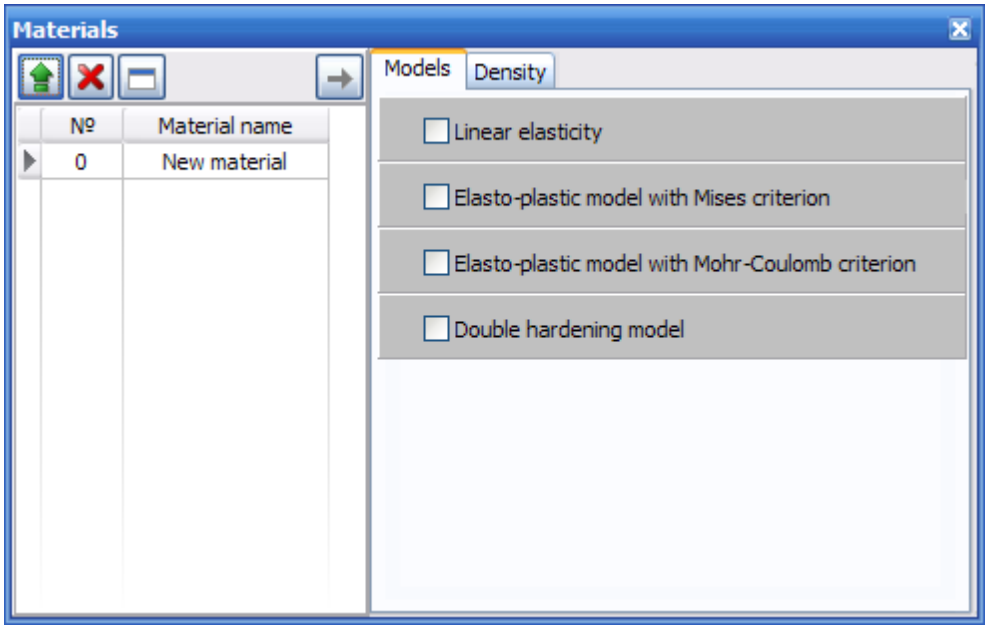


- *Element types*, allows selecting element type (simplex, triangular six-node or beam) and specifying it for a certain area (line) or for all areas (lines).



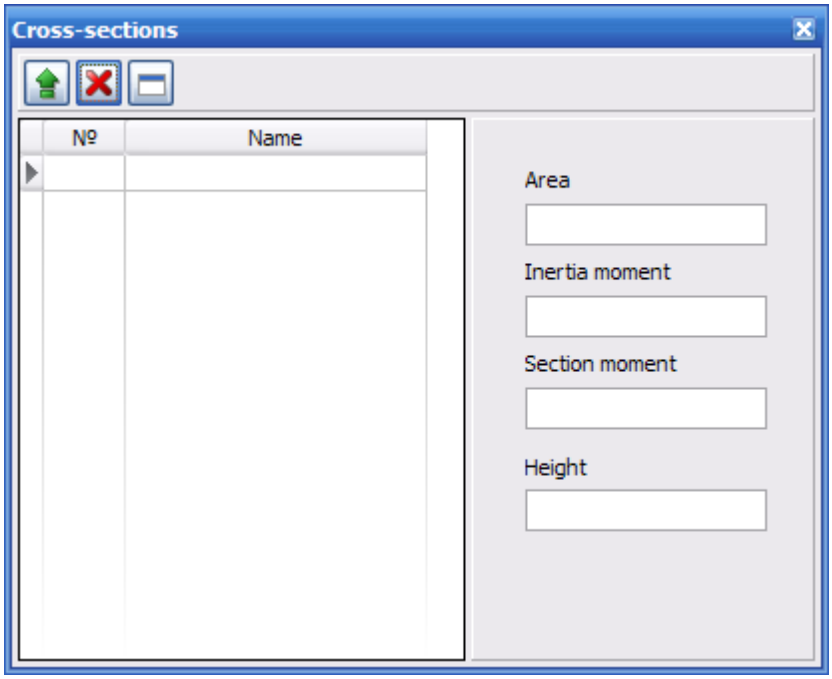
- *Model characteristics*

"Materials" button opens a box for specifying material properties.



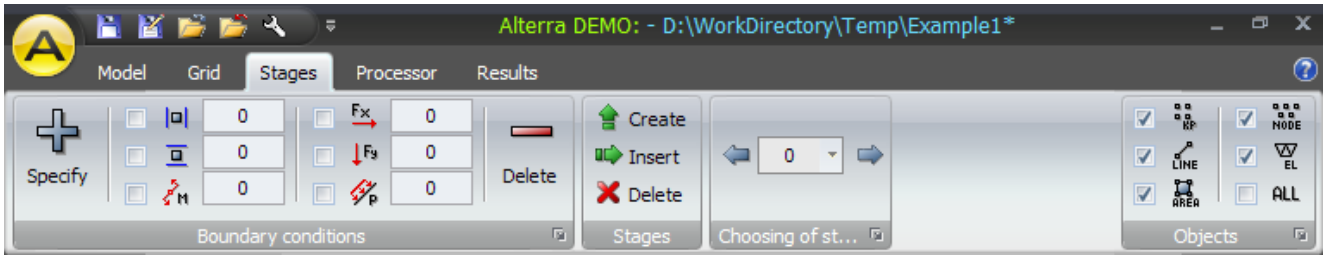
"Cross-sections" button opens a box for specifying beam cross-section properties.

© 2012 Alterra GmbH

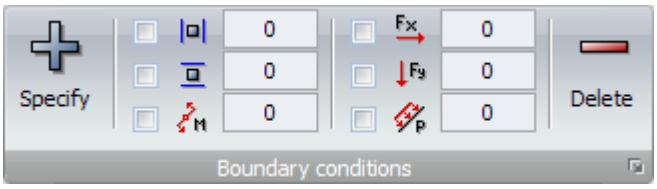


- *Generate mesh*, a button which starts finite element mesh generation.

1. Stages



The page contains the following button groups:



– *Boundary conditions*, allows adding or deleting boundary conditions in lines and points.

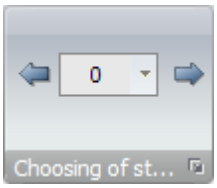
– *Stages*, allows creating or deleting calculation stages.

"Create" button creates a new stage at the end of stage list.

"Insert" button creates a new stage and insert it after a current stage.

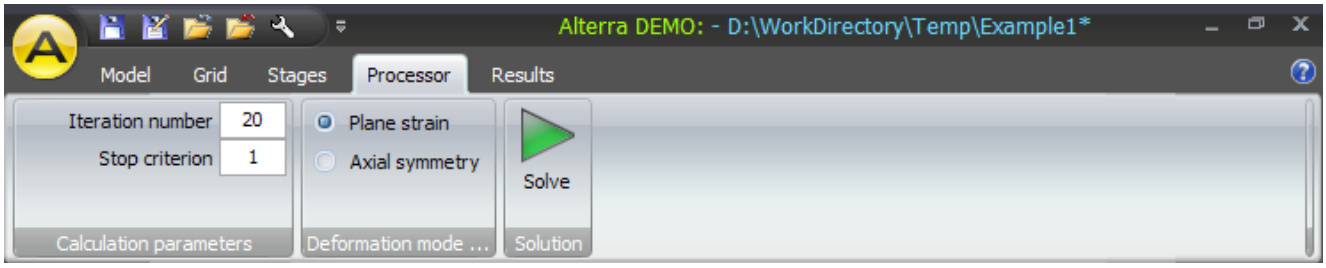
© 2012 Alterra GmbH

"Delete" button deletes a current stage.

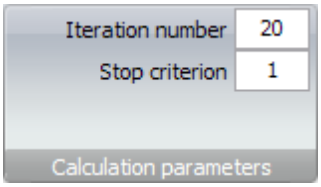


– *Stage navigator*, allows stage list navigating.

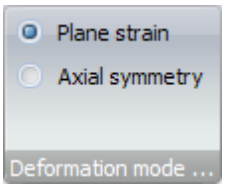
1. Processor



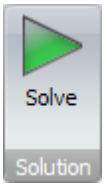
The page contains the following button groups:



– *Calculation parameters*,



– *Deformation mode*, allows selecting type of calculation: plane strain or axial symmetric.

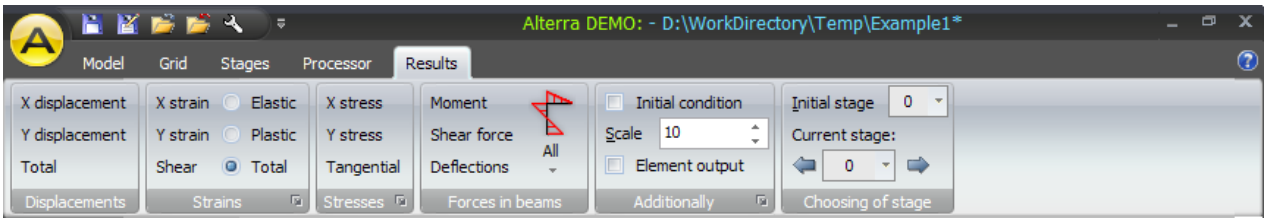


- *Solution*, starts calculation.

1. Results.

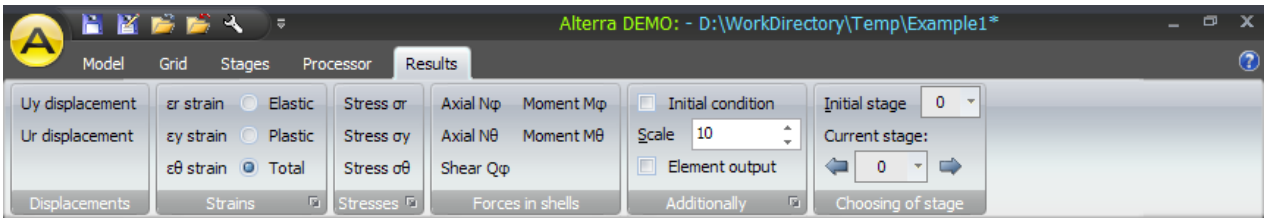
"Results" page view depends on type of calculation.

Plane strain:



© 2012 Alterra GmbH

Axial symmetric:



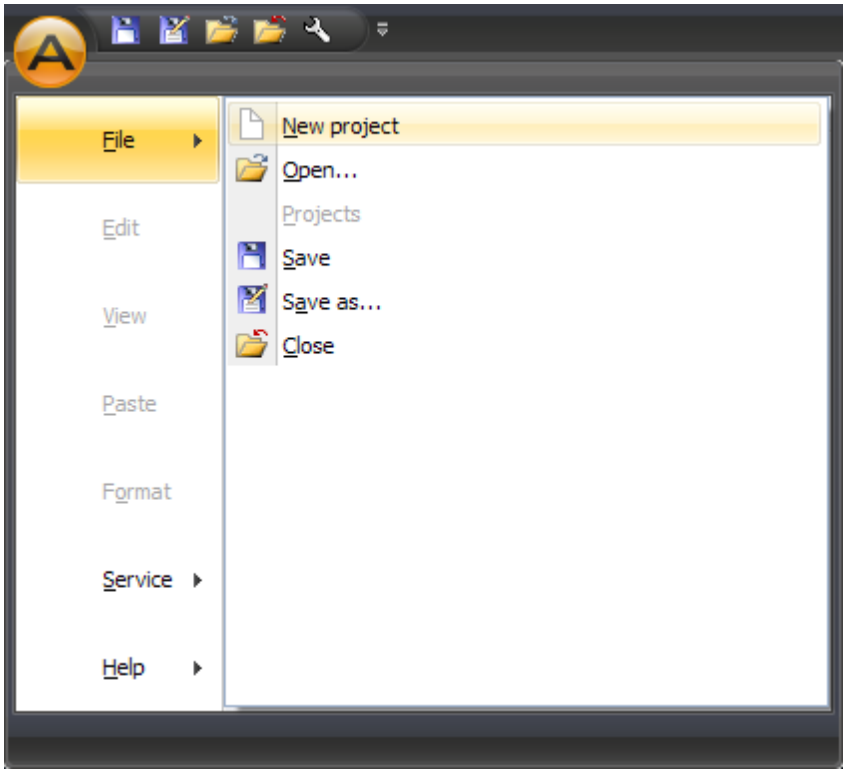
© 2012 Alterra GmbH

Example #1

This example describes a simple cantilevered retaining wall.

1. New project

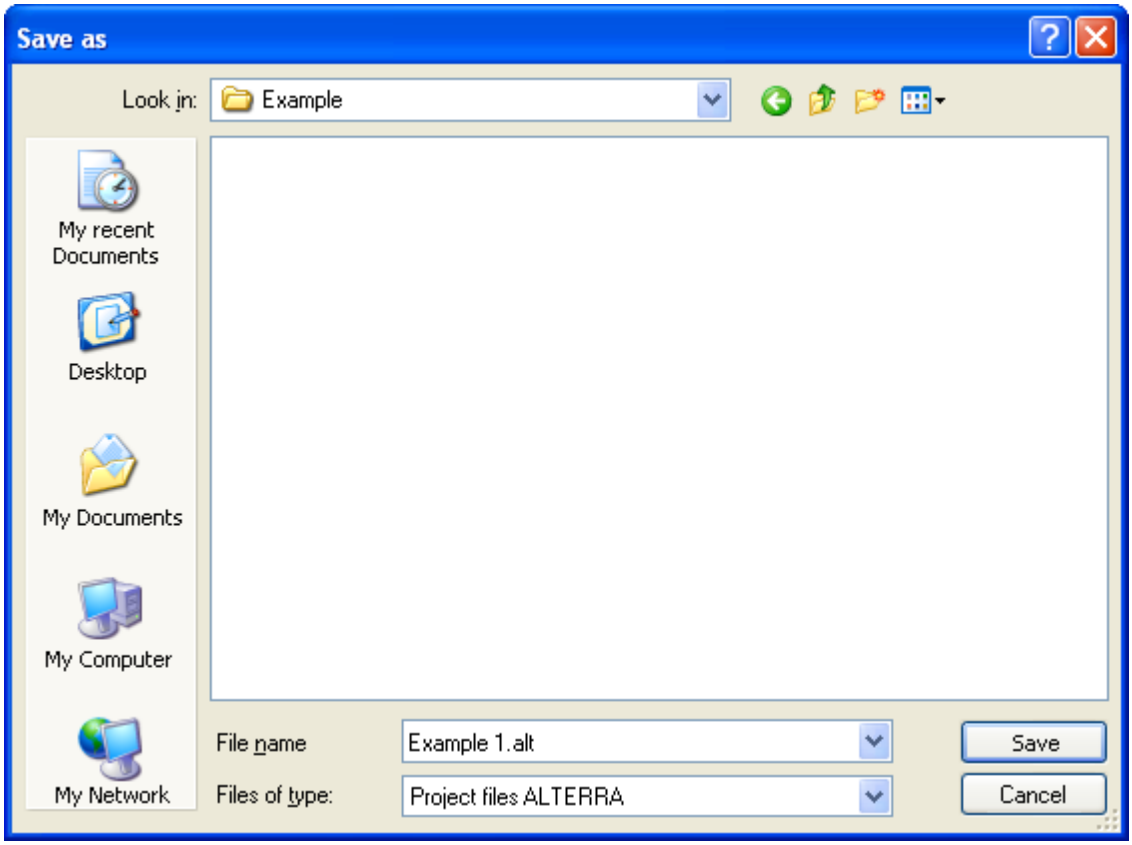
A new project can be created by menu item (Ctrl + N):



"New project" menu item opens a dialogue box in which you should:

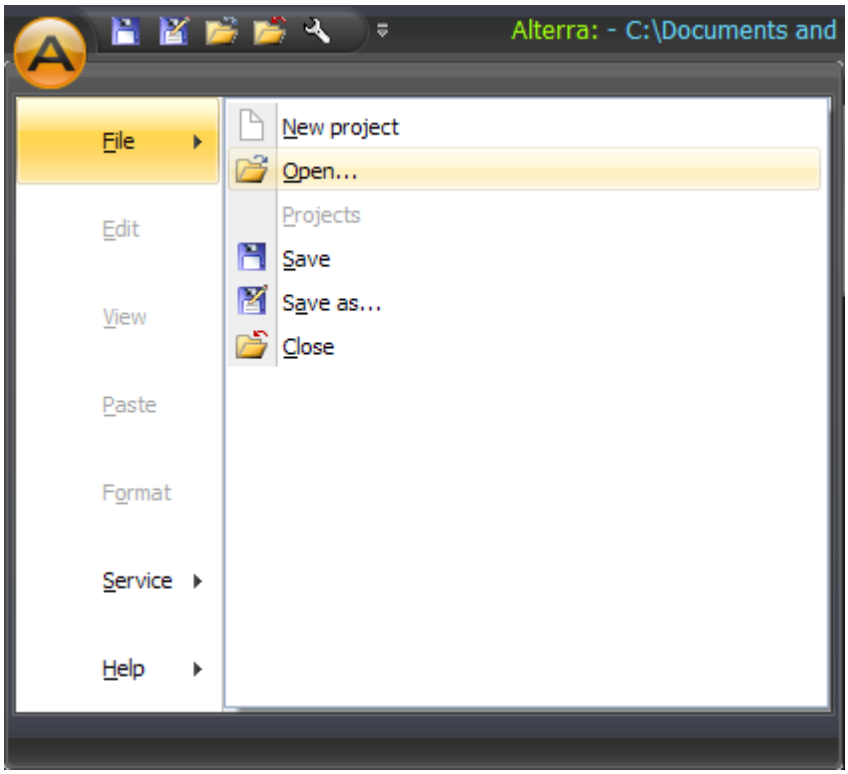
- select a folder to save project files;
- name a project;
- press "Save" button.

© 2012 Alterra GmbH



2. Open project

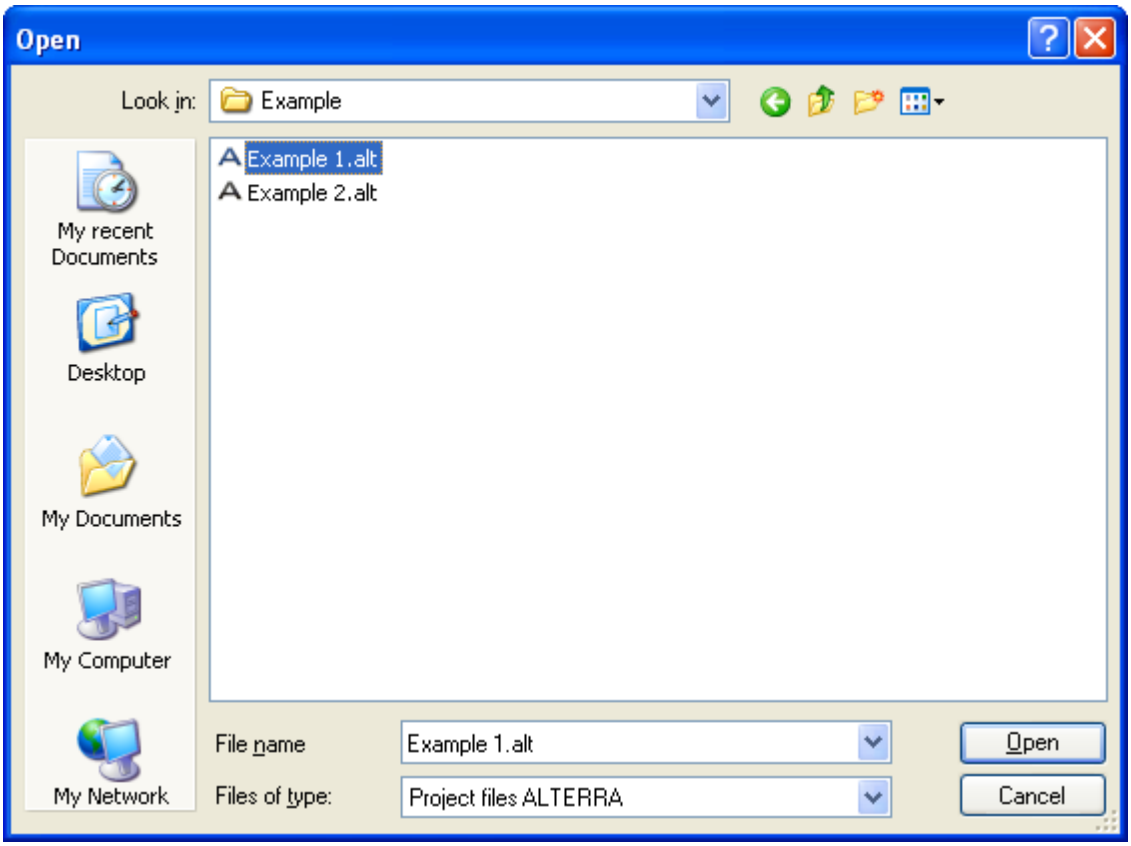
For further work in previously created project it is necessary to open it from "File" submenu (Ctrl + O):



© 2012 Alterra GmbH

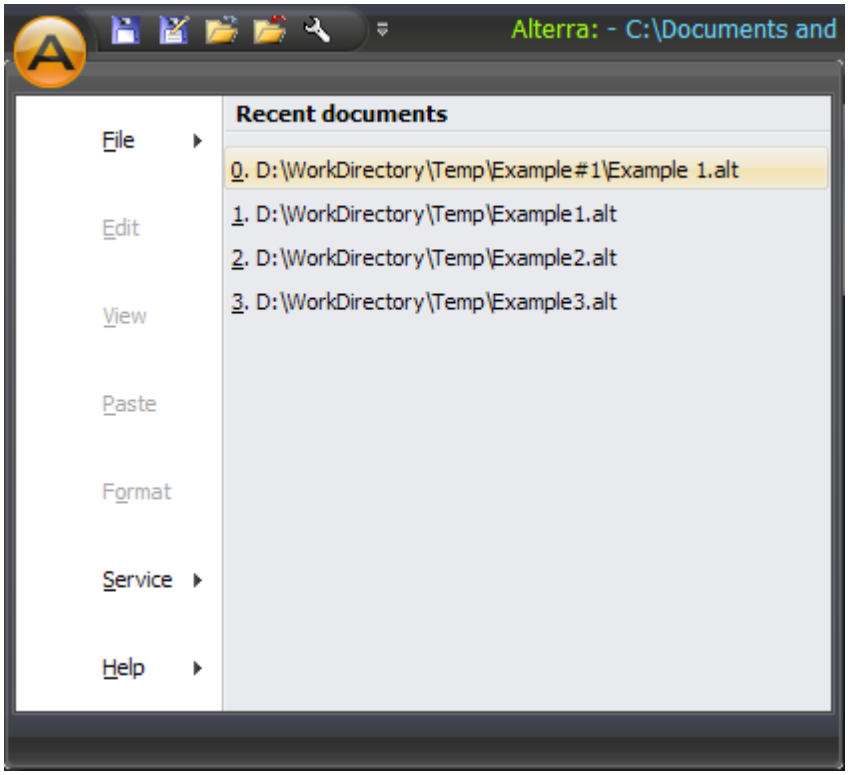
"Open project" menu item opens a dialogue box in which you should:

- select a folder which contains previously created project;
- select a project file;
- press "Open" button.



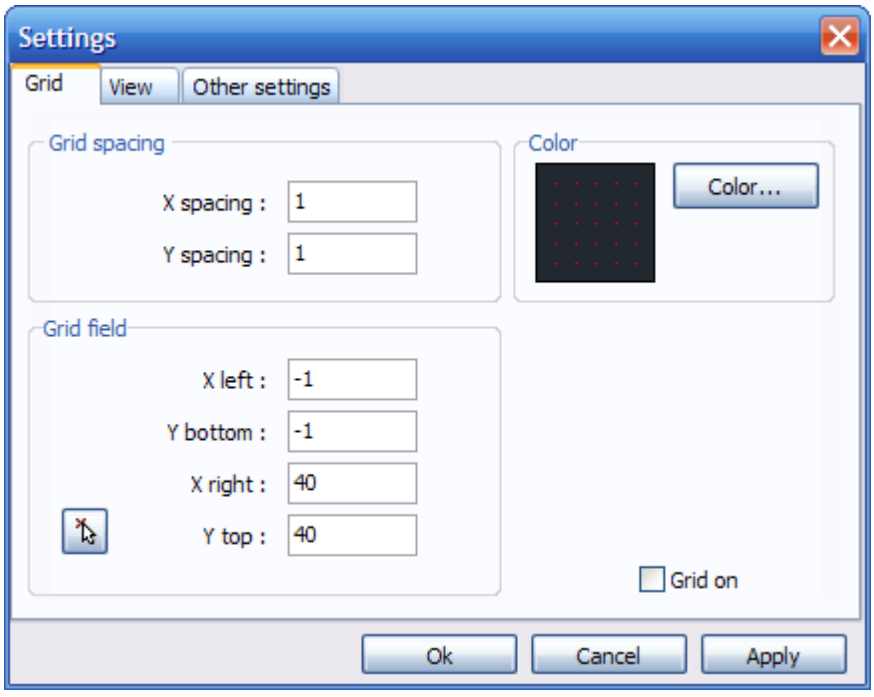
Use "Recent docs" submenu for quick project access.

© 2012 Alterra GmbH



3. Grid option

To create model by mouse there is an option of grid display activated by button (or F9) in status bar. Cursor moves discretely in grid. Grid parameter settings can be specified in "Settings" box on "Grid" page.



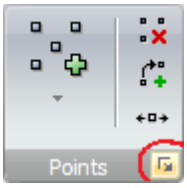
© 2012 Alterra GmbH

4. Point adding

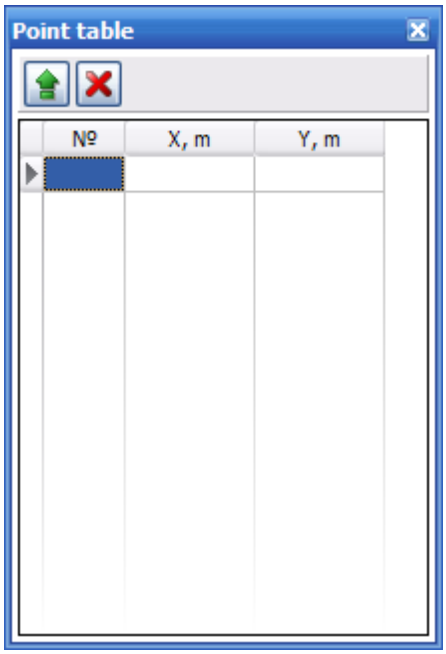
There are three ways to add points:

1. Move to "Model" page, press button in "Points" group and add points in input field by mouse.

2. Press "Additional" button on "Model" page



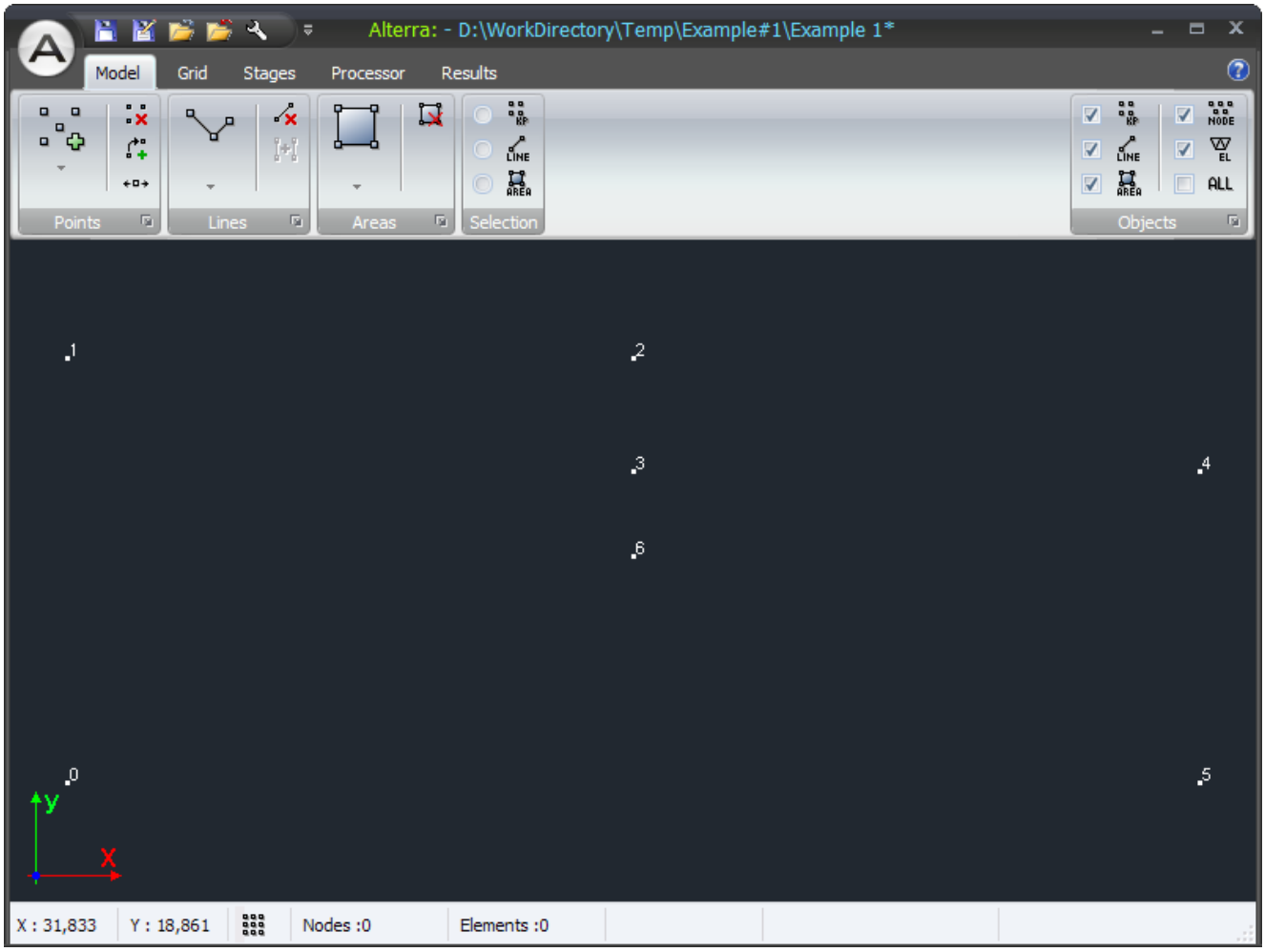
In appeared "Point table" box input point numbers and point coordinates manually.



3. Import geometry from DXF file. It is necessary that DXF file contains only points and lines (not polylines).

When points are added input field has the following view:

© 2012 Alterra GmbH



5. Point deleting

There are three ways to delete points:

1. Press button in "Points" group on "Model" page, then select a point by mouse and press LMB.

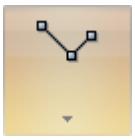
2. Select a row in "Point table" box and press button.

3. Check a point selection item in "Selection" group, then select points by frame pressing LMB and press Delete.

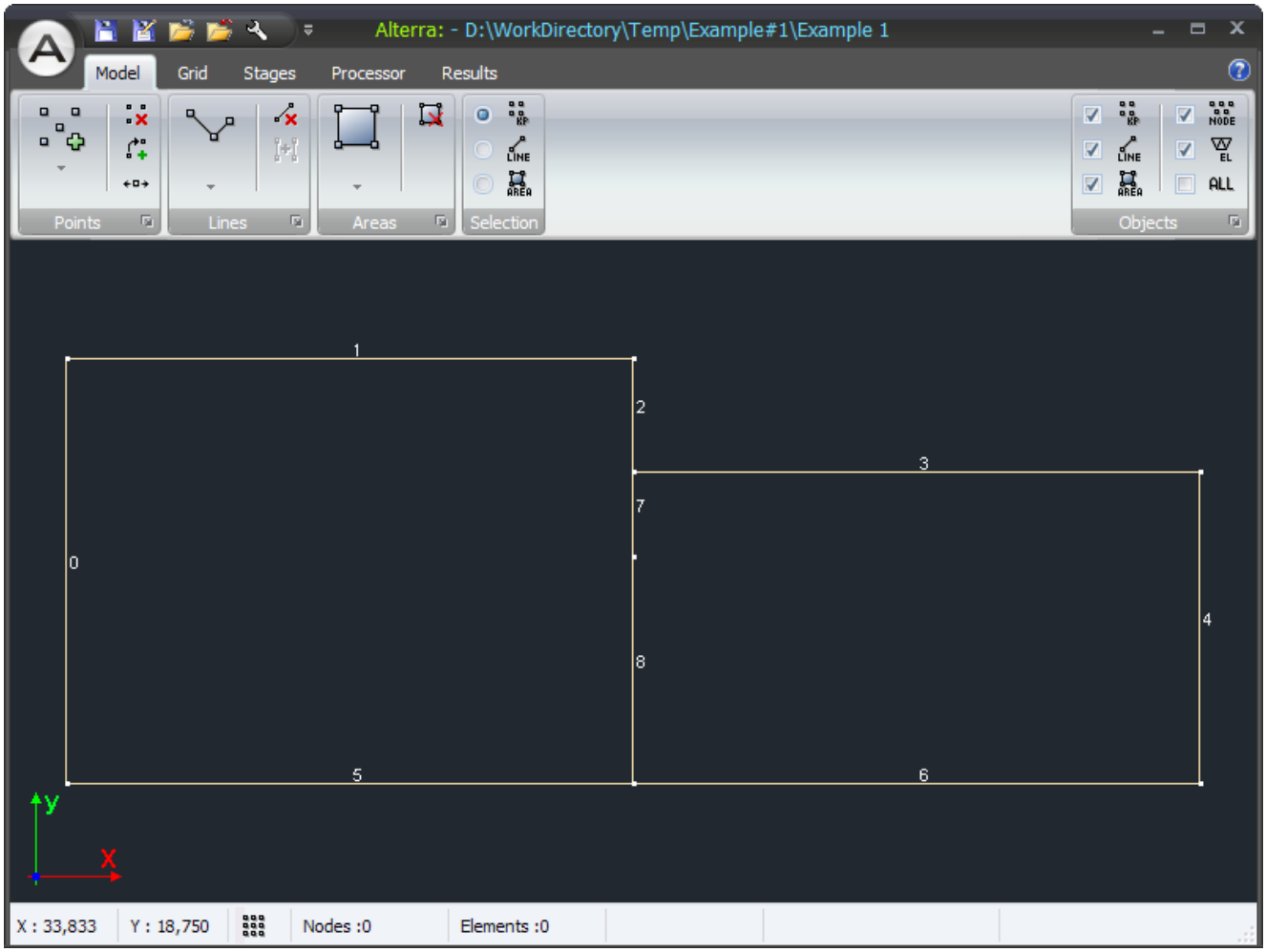
© 2012 Alterra GmbH

6. Line creating

There are two way to create lines:

1. Press button in "Lines" group on "Model" page, then join points by LMB. A previous line end is a next line origin. To complete polylines creation press RMB or uncheck  button.

When lines are added input field has the following view:



© 2012 Alterra GmbH

7. Line deleting

There are two ways to delete lines:

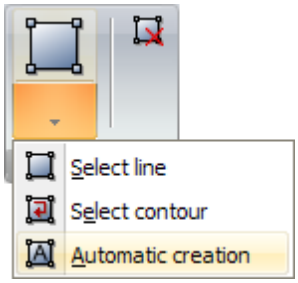
1. Press button in "Lines" group on "Model" page, then locate cursor on line and press LMB.

2. Check a line selection item in "Selection" group, then select lines by frame pressing LMB and press Delete.

8. Area creating

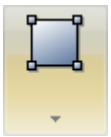
There few ways to create areas:

1. Press button in "Areas" group on "Model" page. All closed convex areas will be found automatically.



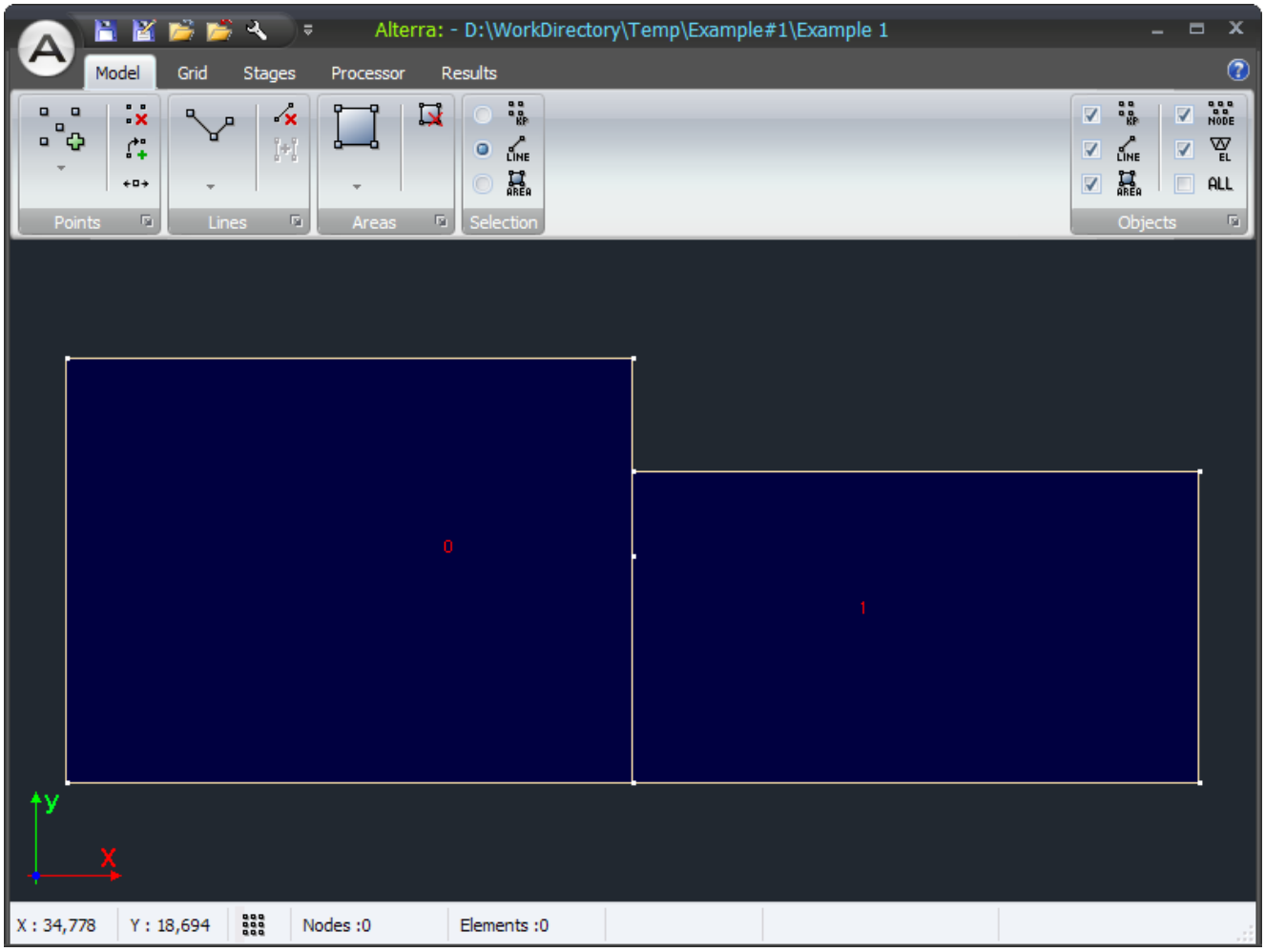
Note:

If some areas are not created automatically it is necessary to create them manually.

2. Press  button in "Areas" group on "Model" page. Select lines by LMB and press RMB. These lines should form closed convex area.

When areas are added input field has the following view:

© 2012 Alterra GmbH



9. Area deleting

There are two ways to delete areas:

1. Press , button in "Areas" group on "Model" page, then locate cursor on line and press LMB.

2. Check an area selection item in "Selection" group, then select areas by frame pressing LMB and press Delete.

© 2012 Alterra GmbH

10. Physical-mechanical properties of materials

Press in "Material properties" box to create a new material.

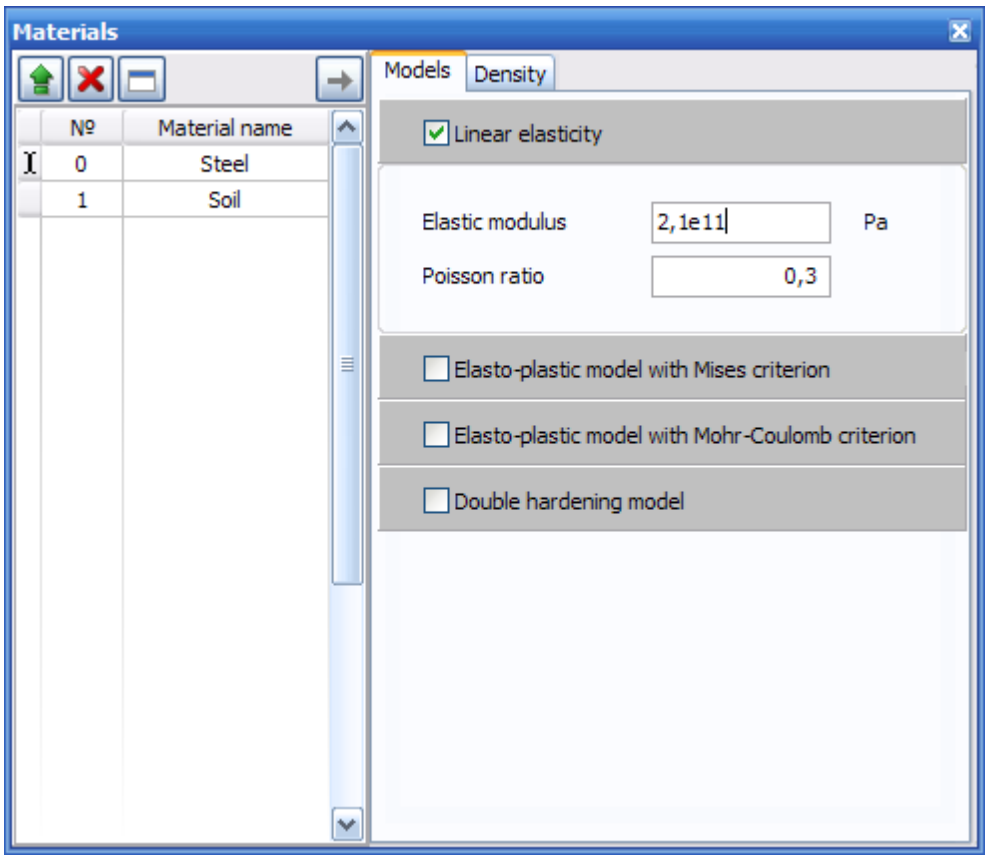
Press button to delete material.

Let us add two materials such as "steel" and "soil".

Choose linear elastic model for steel. The material have the following properties:

E = 210 [GPa] - elasticity modulus of steel

ν = 0,3 - Poisson ratio



Choose elastoplastic (Mohr-Coulomb yield criterion) material model for soil. The material have the following properties:

c = 8 [kPa] - soil cohesion

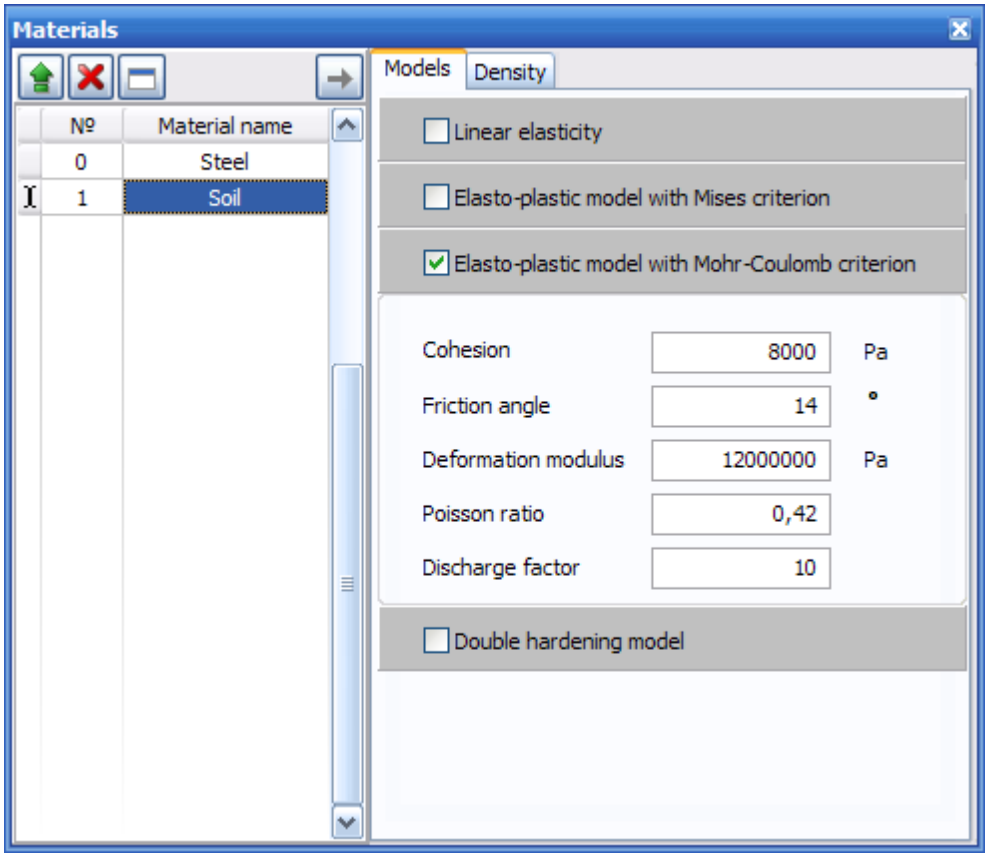
ϕ = 14 [°] - internal friction angle

E = 12 [MPa] - deformation modulus of soil

ν = 0,42 - Poisson ratio

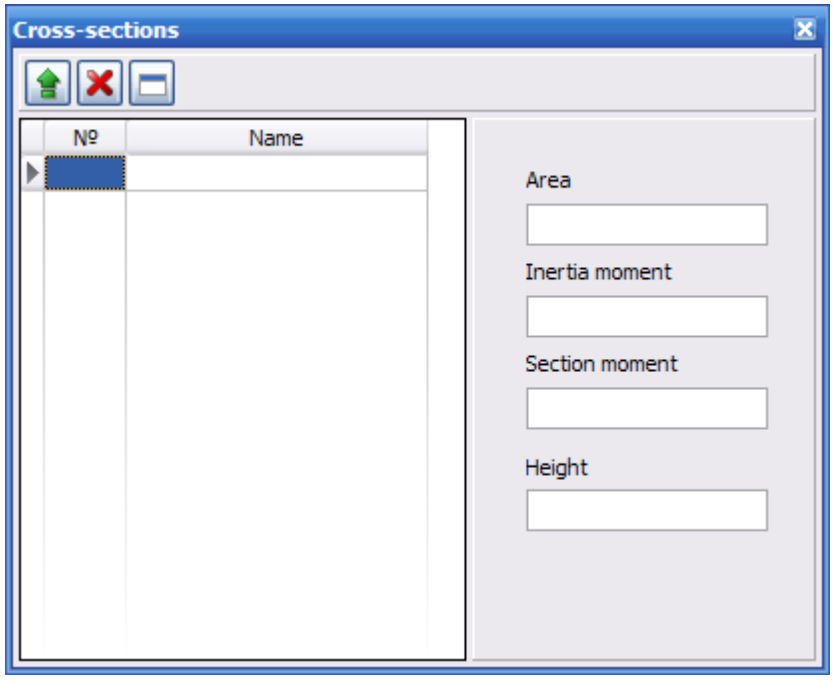
k = 10 - unloading ratio

© 2012 Alterra GmbH



11. Cross-section properties

Beam cross-section properties are specified in the following box:



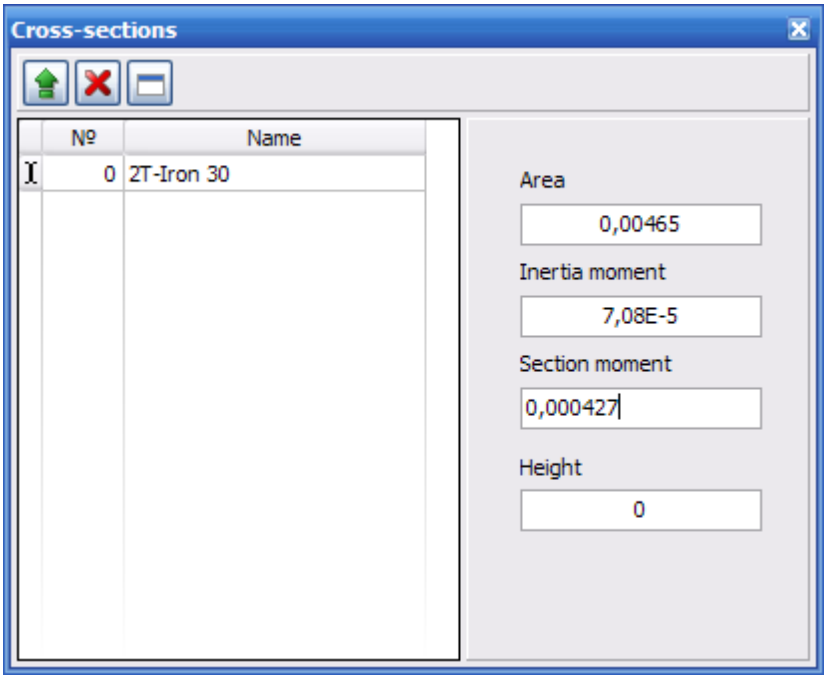
Retaining wall consists of I-beam 30 with 1 m spacing:

S = 0.00465 [m^2] - cross section area

© 2012 Alterra GmbH

I = 0.708*10^{-4} [m^4] - cross section inertia moment

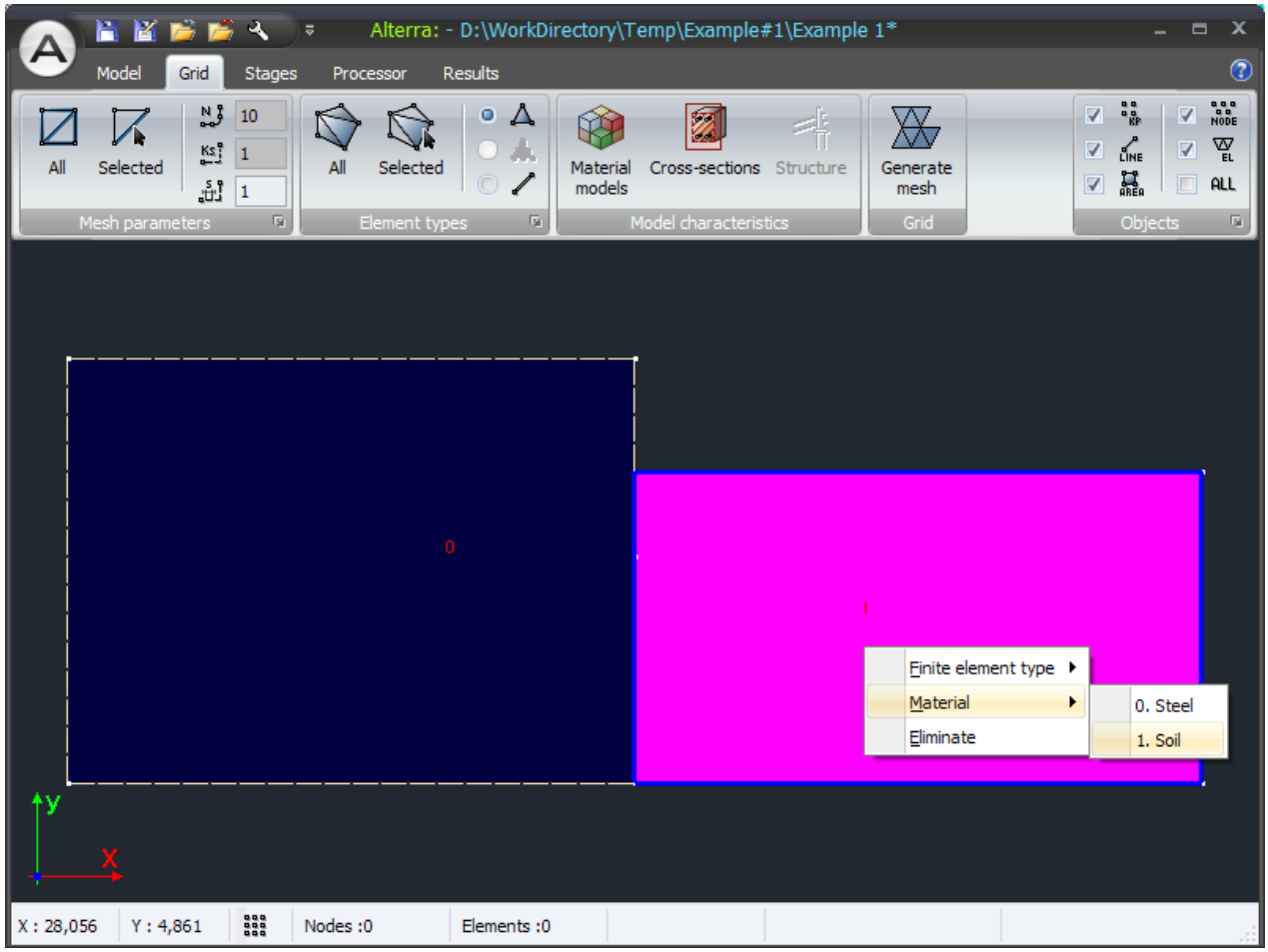
It is not necessary to specify "Cross section thickness" if you specify area and inertia moment of cross section.



12. Material property specifying

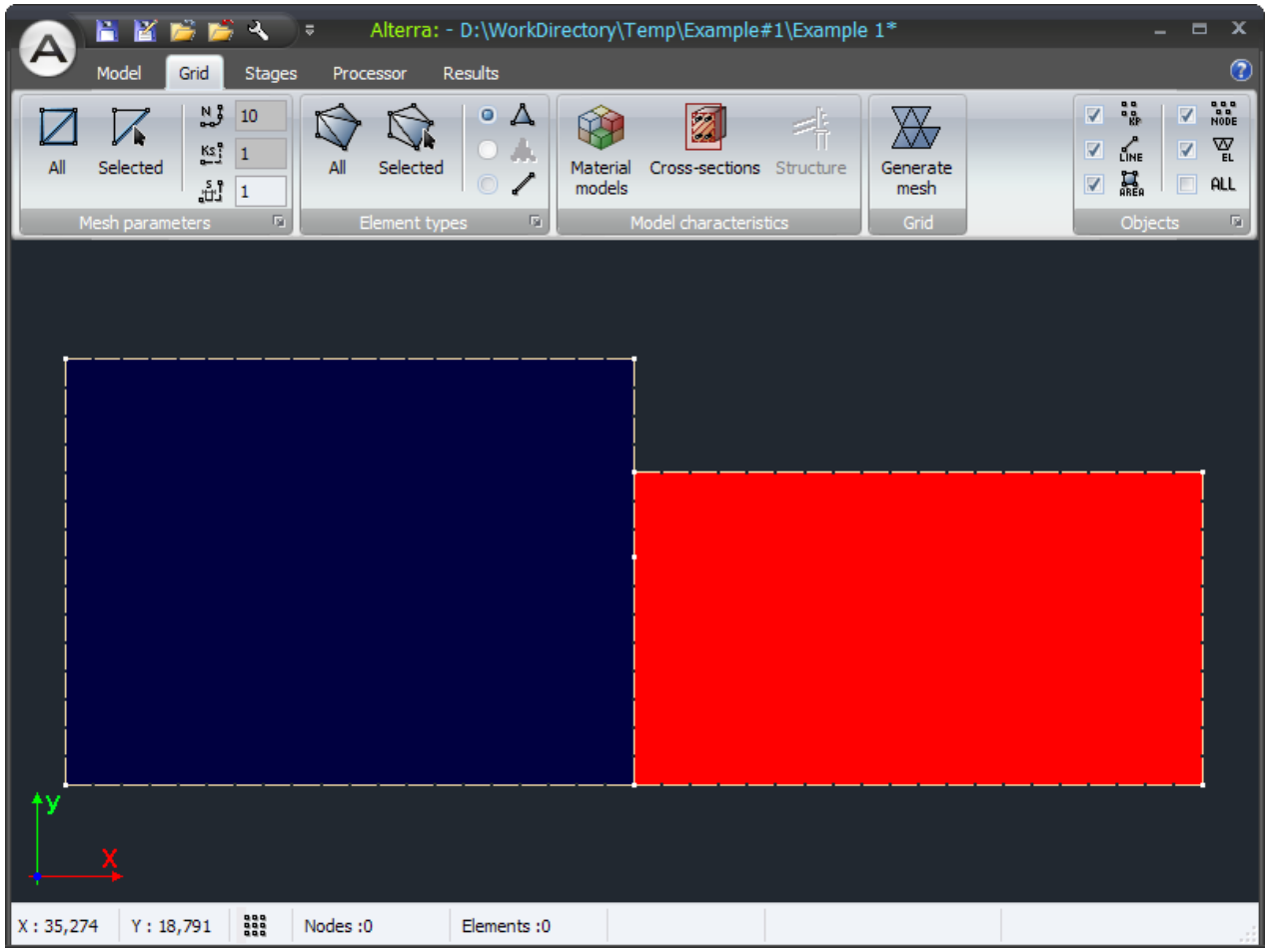
Select an area by double click (this area will change its color). Press RMB and select "Material" -> "Soil" points in pop-up menu.

© 2012 Alterra GmbH



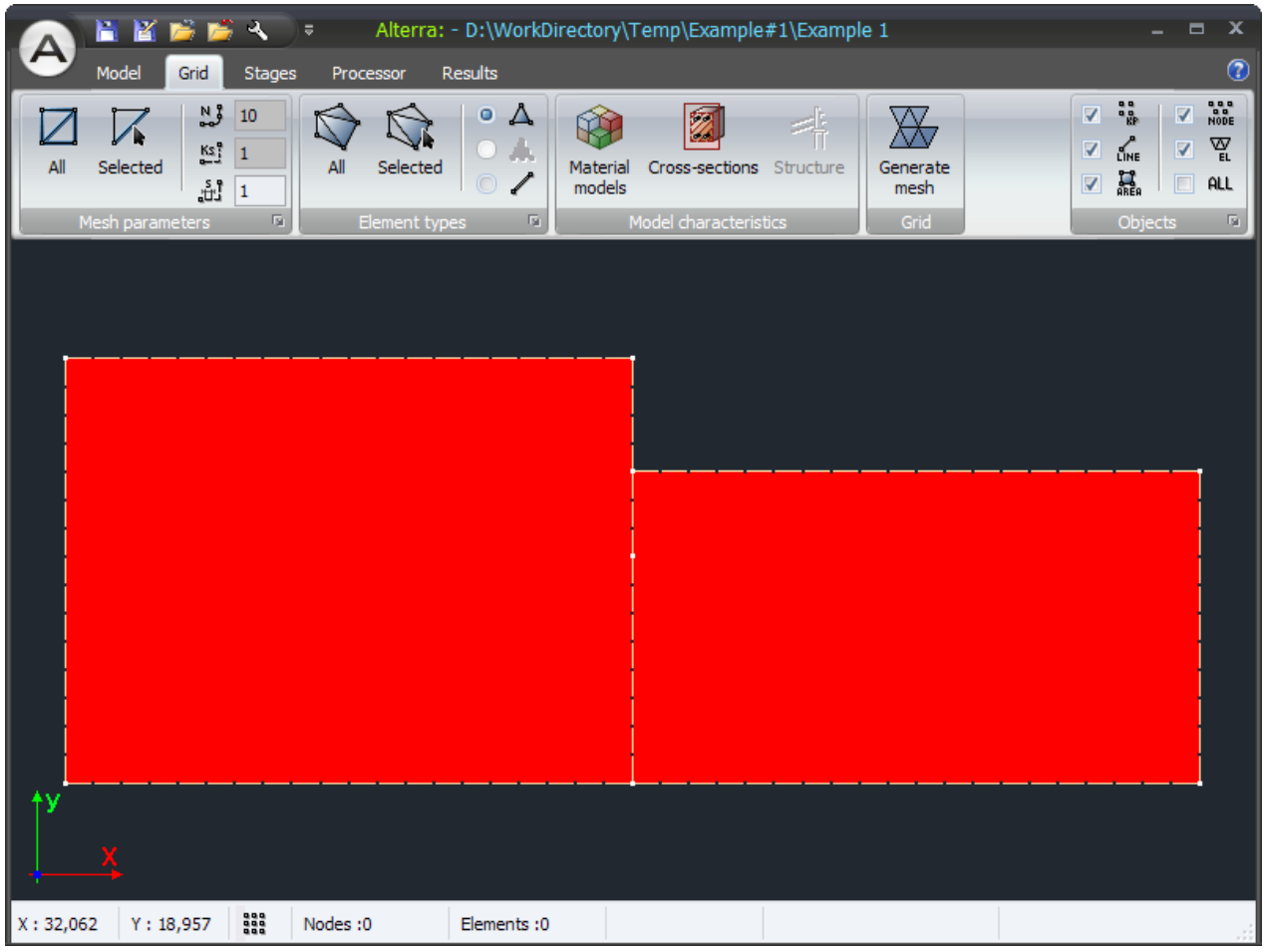
After specifying selected area changes its color which corresponds to specified material.

© 2012 Alterra GmbH



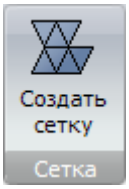
Material for the next area and lines (which simulate retaining wall) is specified in the same way.

© 2012 Alterra GmbH



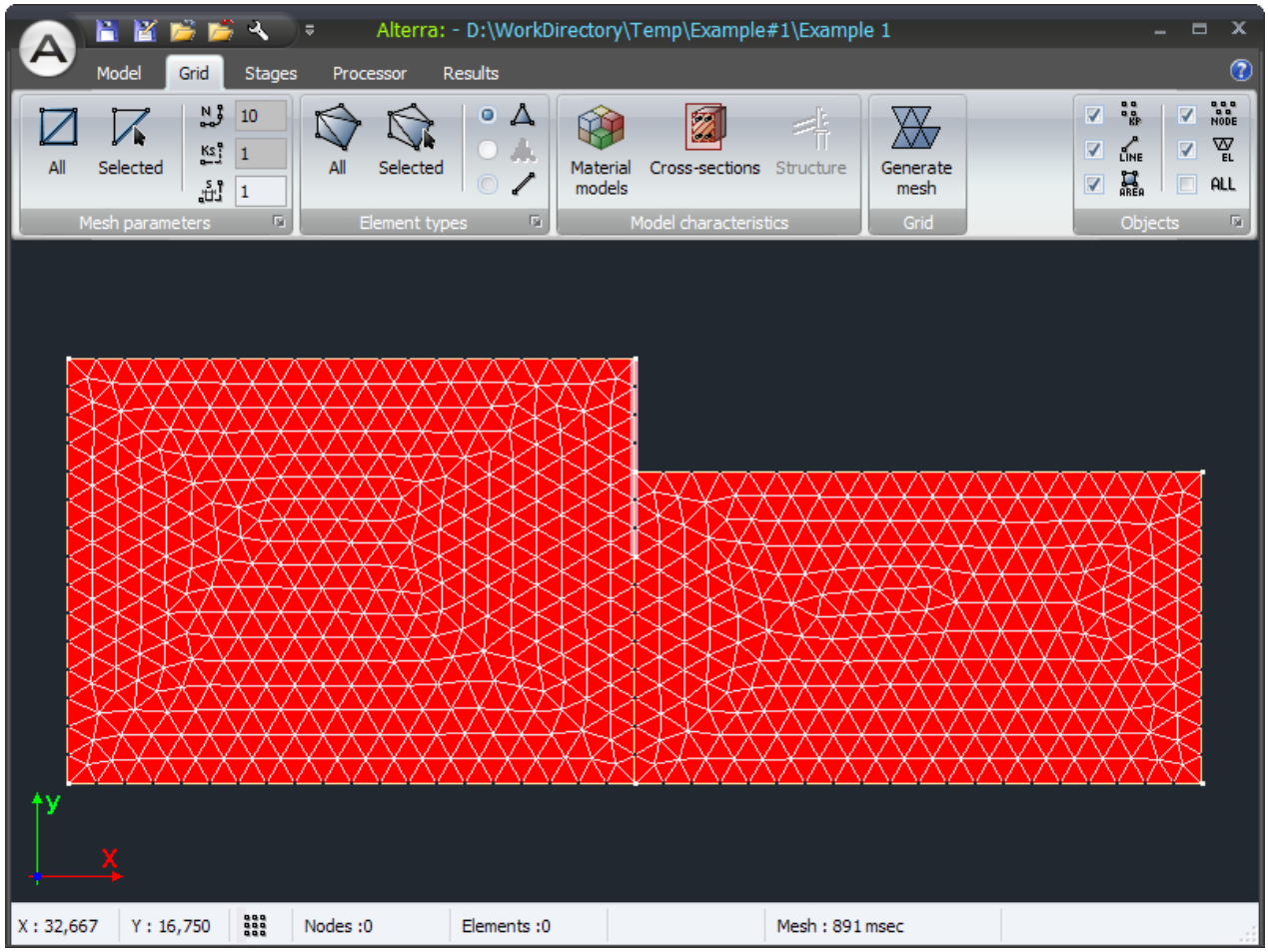
13. Mesh creating

When element size for all lines and element type for all areas are specified press "Create a mesh" button in "Mesh" group on "Mesh" page:



After that the model is meshed into finite elements, in our case they are simplex element simulating soil and beam element simulating retaining wall.

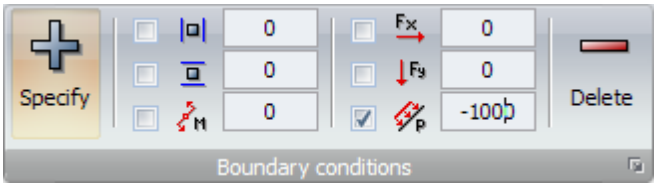
© 2012 Alterra GmbH



14. Boundary conditions

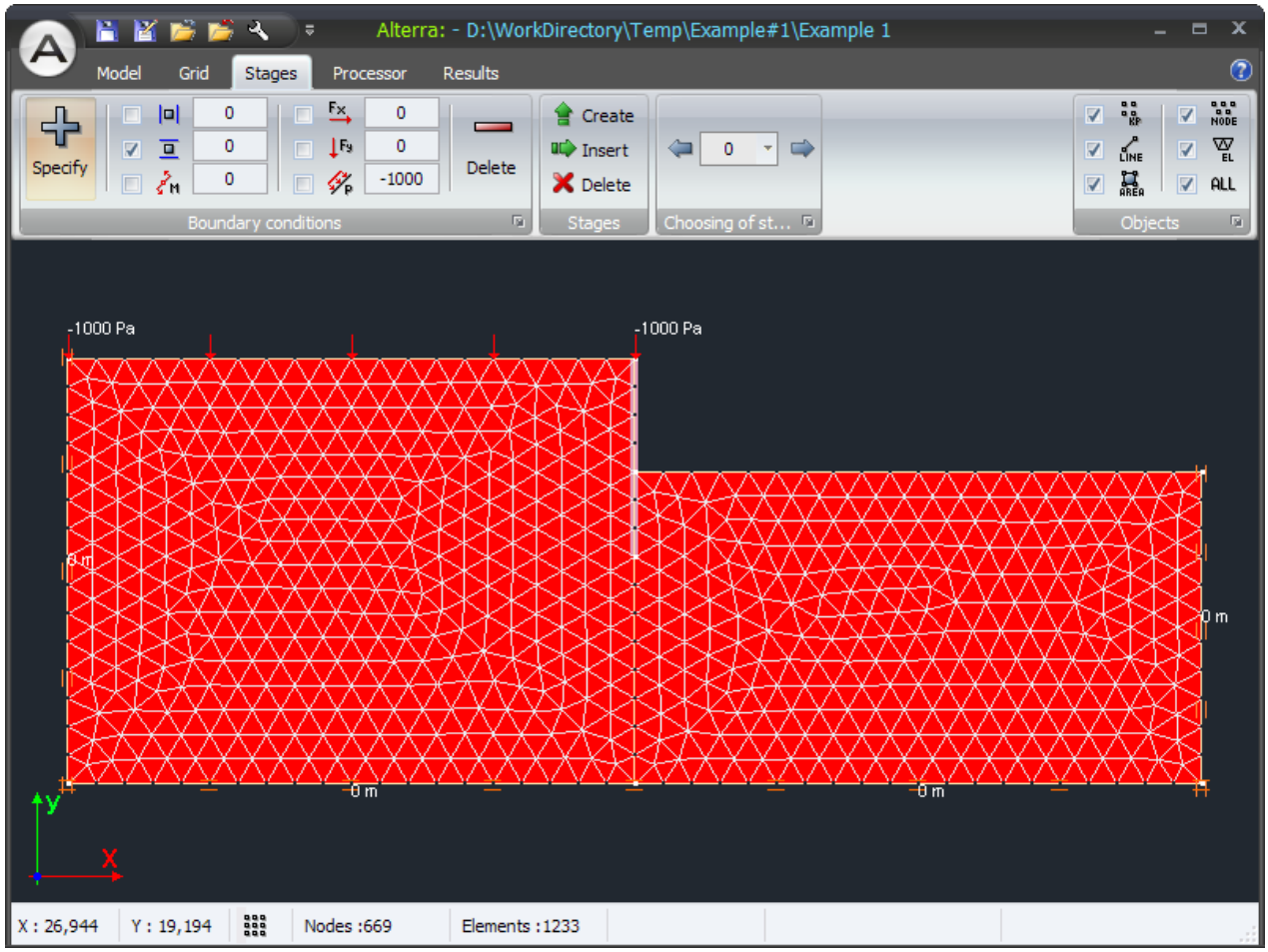
It is possible to specify loads or displacements as boundary conditions.

Using "Boundary conditions" group on "Stages" page:



Specify boundary conditions as follows:

© 2012 Alterra GmbH



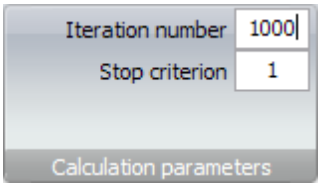
15. Calculation

When finite element mesh is created and all materials are specified let us pass to "Solver" page.

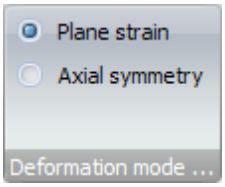
Input two parameters in "Calculation parameters" group:

n = 1000 - iteration number - max iteration number per one stage

k = 1 [H] - stop criterion - max unbalanced force per one stage

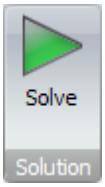


Select "Plane strain" point in "Deformation mode" group



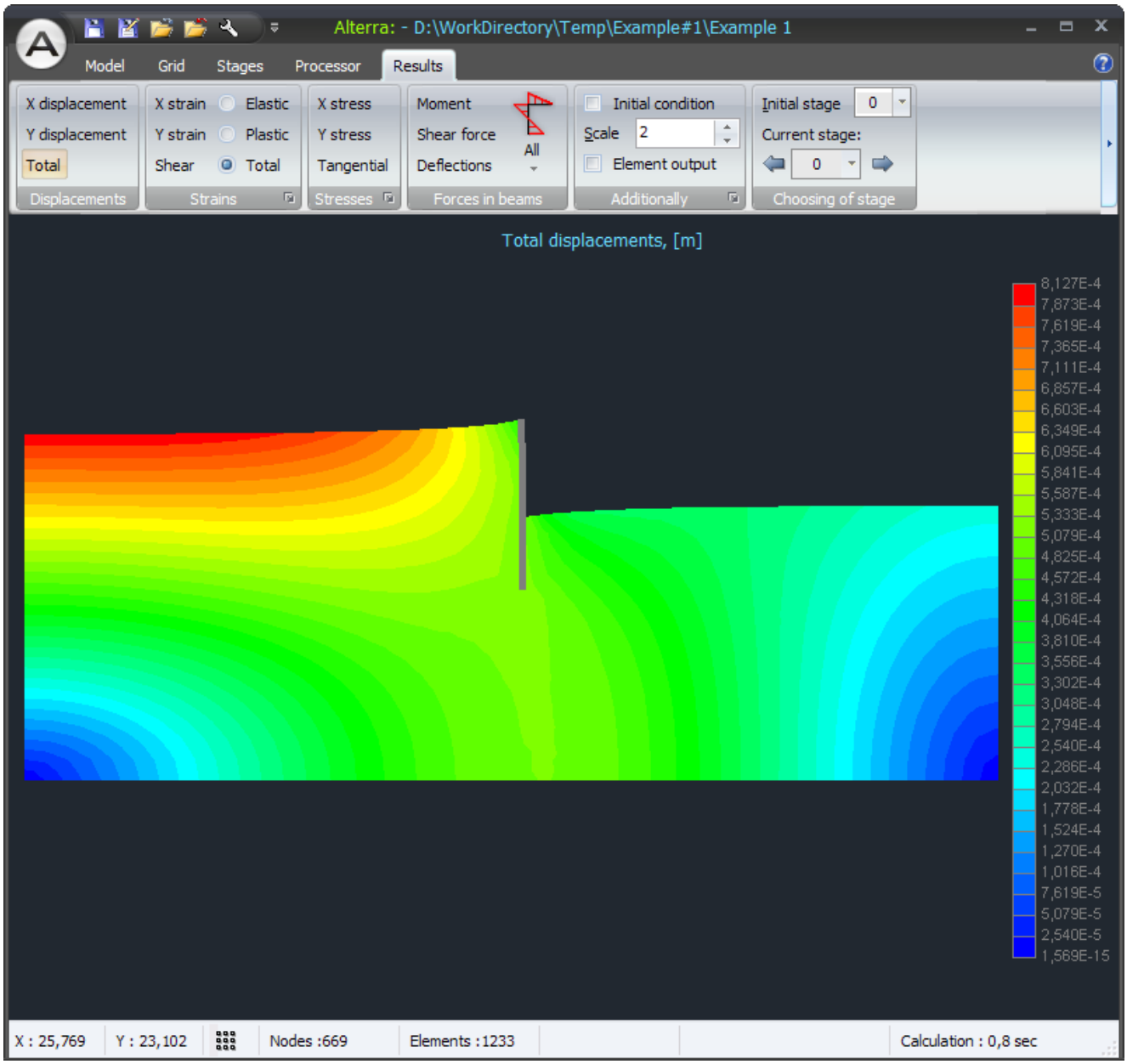
Press "Calculation" button

© 2012 Alterra GmbH



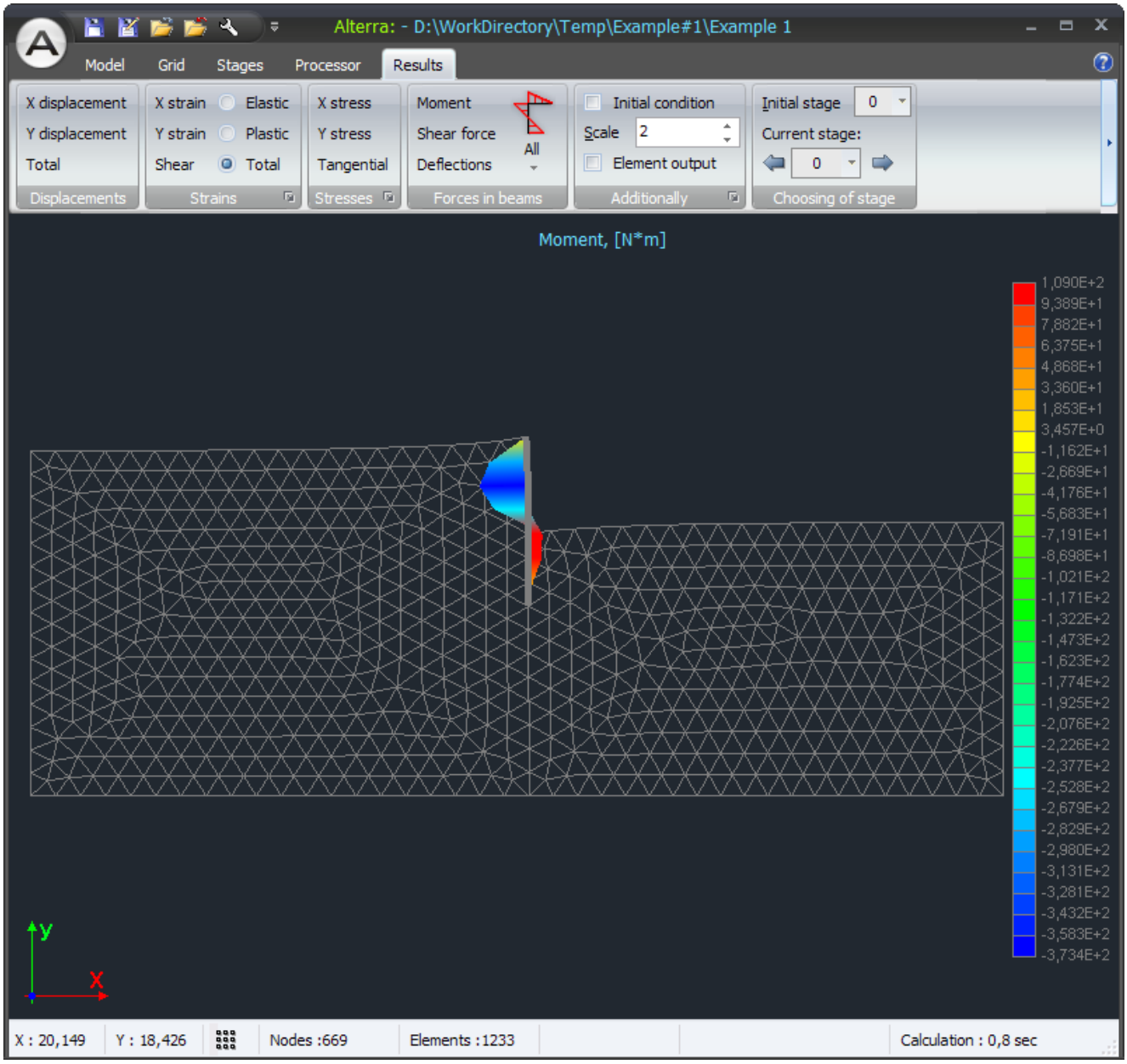
16. Result output

The program allows estimating stress and strain behaviour of soil:



and retaining wall

© 2012 Alterra GmbH



© 2012 Alterra GmbH